SOLIDWORKS Sheet Metal Professional (CSWP-SM) Certification Prep Course

Target Audience

This course is designed for SOLIDWORKS users who want to develop sheet metal design skills and prepare for the CSWP-SM certification. It is ideal for professionals, engineers, and designers who already have basic SOLIDWORKS knowledge and aim to enhance their expertise in sheet metal functionalities, standardized workflows, and advanced techniques required to successfully pass the certification exam.

Course Outcomes

- Understand sheet metal design concepts and workflows within SOLIDWORKS
- Apply flange creation and flat pattern techniques effectively
- Create and modify sheet metal parts using standard and advanced features
- Optimize sheet metal designs using standardization methods and tables
- Troubleshoot issues in flat patterns, conversions, and multibody designs
- Prepare for the CSWP-SM certification with practical and exam-oriented knowledge

Course Objectives

- Develop structured knowledge of sheet metal design tools and features in SOLIDWORKS
- Build proficiency in creating and editing sheet metal components
- Enable learners to work with flat patterns and manufacturing-ready outputs
- Introduce advanced techniques such as lofted bends, multibody parts, and forming tools
- Reinforce learning through hands-on exercises aligned with certification requirements
- Prepare participants for the CSWP-SM certification exam with comprehensive coverage

Course Outline

The course comprises **40-hours** of theory and labs and is divided into **11** different chapters. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered.

Table of Contents (TOC)

- Invoking the Part Modeling Environment
- Invoking the Assembly Environment
- Invoking the Drawing Environment
- Identifying SOLIDWORKS Documents
- Customizing the Context Toolbar
- Customizing the Command Manager
- Working with Mouse Gestures
- Saving Documents
- Opening Existing Documents

Module 2: Sketching Fundamentals

- Specifying Units
- Invoking the Sketching Environment
- Working with the Selection of Planes
- Specifying Grids and Snap Settings
- Drawing a Line
- Drawing a Rectangle
- Drawing a Circle
- Drawing an Arc
- Drawing an Ellipse
- Drawing a Spline
- Trim
- Offset
- Convert Entities
- Sketch Relations
- Smart Dimension
- Fully Defined Sketch

Module 3: Base Features and Part Modeling

- Creating an Extruded Feature
- Creating a Revolved Feature
- Creating Cut Features
- Fillet
- Chamfer
- Shell
- Draft
- Rib
- Hole Wizard
- Linear Pattern
- Circular Pattern
- Mirror
- Navigating a 3D Model
- Manipulating View Orientation
- Changing Display Style

Module 4: Basic Flange Features

- What are Sheet Metal Parts
- Sheet Metal Methods
- Unique Sheet Metal Items
- Flange Method
- Base Flange/Tab
- Sheet Metal Parameters
- Sheet Metal Thickness and Bend Radius
- Bend Allowance
- K-Factor
- Bend Allowance
- Bend Deduction
- Specifying the Bend Allowance
- Auto Relief
- Editing SheetMetal Parameters
- Sheet Metal Bend Features
- Flat-Pattern Feature
- Flatten and Exit Flatten
- Toggle Flat Display
- Additional Flange Features
- Edge Flanges
- Edge Flange Settings
- Editing the Flange Profile
- Flange Profile Relations
- Edge Flanges on Curved Edges
- Miter Flanges
- Miter Flange Settings
- Hem Feature
- Hem Settings
- Tab Features
- Cuts in Sheet Metal
- Sharing a Model
- Summary of Flange Features

Module 5: Working with the Flat Pattern

- Working with the Flat Pattern
- Flat Pattern Settings
- Features for Manufacture
- Bend Notch Feature
- Corner-Trim Feature
- Corner-Trim Settings
- Corners in the Formed State
- Closed Corner
- Closed Corner Settings
- Corner Relief

- Break Corner/Corner Trim
- Producing the Flat Pattern
- Sheet Metal Cut List Properties
- Accessing Cut List Properties
- Sheet Metal Drawings
- Flat Pattern Drawing Views
- Flat Pattern View Properties
- Cut List Properties as a Note
- Exporting the Flat Pattern

Module 6: Standardizing Sheet Metal Designs

- Standardizing Gauge Numbers and Bend Radii
- Standardizing Bend Allowance
- Topics for Standardizing Parameters
- Using Tables
- Gauge Tables
- Bend Tables
- Bend Allowance
- Editing the Gauge Table Selection
- Gauge Table Training Files
- Defining Table File Locations
- Custom Sheet Metal Materials
- Sheet Metal Templates
- Sheet Metal Part Document Properties
- Other Part Template Settings
- Sensors for Sheet Metal
- Sheet Metal Drawing Document Properties
- Sheet Metal Tables in Drawings
- Adding a Cut List Table
- Adding a Bend Table
- Mapping DXF Output
- Options for Map File

Module 7: Additional Sheet Metal Techniques

- Additional Sheet Metal Methods
- Designing from the Flat
- Sketched Bend Feature
- Jog Feature
- Adding Features in an Unfolded State
- Unfold and Fold
- Creating Cuts in the Flat Pattern
- Swept Flange
- Swept Flange Flat Pattern Options
- Lofted Bends
- Bent Lofted Bends

- Bent Bend Region Options
- Formed Lofted Bends
- Formed Bend Region Options
- Lofted Bends in the Design Library
- Manual Relief Cut
- Sheet Metal Library Features

Module 8: Converting to Sheet Metal

- Sheet Metal Conversion
- Insert Bends Method
- Adding Rips
- Insert Bends
- Associated Features
- Switching Between States
- Making Changes
- Welded Corner
- Converting Cones and Cylinders
- Convert to Sheet Metal
- Convert to Sheet Metal Settings
- Using Rip Sketches

Module 9: Multibody Sheet Metal Parts

- Multibody Sheet Metal Parts
- Tools to Create Multibody Sheet Metal Parts
- Multibodies with Base Flange
- Sheet Metal Parameters for Multibodies
- Solid Body Feature History
- Cut List Item Properties for Multibodies
- Flat Pattern Drawing Views for Multibodies
- Cut List Balloon Annotations
- Exporting to DXF/DWGs with Multibodies
- Convert with Multibodies
- Hiding and Showing Bodies
- Hide and Show
- Hide/Show Bodies Command
- Isolate
- The Display Pane
- Sensors for Multibody Parts
- Using Split with Sheet Metal Parts
- Patterning for Multibodies
- Using Edge Flanges to Merge Bodies
- Interfering Bodies
- Combining Sheet Metal with Other Bodies
- Assigning Materials to Bodies

Module 10: Forming Tools and Gussets

- Sheet Metal Forming Tools
- How They Work
- Forming Tools in the Design Library
- The Forming Tools Folder
- Using an Existing Forming Tool
- Form Tool Feature Settings
- Form Tool Features in the Flat
- Part Document Properties
- Custom Forming Tools
- Split Line
- FormingTool Feature
- Legacy Behavior for Forming Tools
- Form Tools in Drawings
- Punch Tables and Punch ID
- Sheet Metal Stamp
- Sheet Metal Gusset

Module 11: Additional Sheet Metal Functions

- Additional Sheet Metal Functions
- Cross-Breaks
- Cross Break Settings
- Cross Breaks in Drawings
- Vent Features
- Fill Pattern
- Mirror Part
- Tab and Slot
- Process Plans